Journal of University of Babylon, Engineering Sciences, Vol.(26), No.(4): 2018.

Fatigue Analysis of Welding Joints of ASTM A36 Low Carbon Steel by Using Finite Element Method

Manar Abdulameer AL-Musawi
Babylon University
Metallurgical Engineering
Diploma In Casting and Welding Metallurgy
manar44manar44@gmail.com

Abdul Rheem. K. Abid Ali
Babylon University
College of Materials Engineering
Metallurgical Engineering
abidalieng@yahoo.com

Abstract

In this study, fatigue was analyzed for several types of weld joints, including the Butt joints, pipe-flange Butt joint, Lap and Tee joint. These weldments were studied under increasing loads by using CATIA software for modeling and using the ABAQUS environment to analyze the resulting stresses. Where the study included arc welding process, welded parts and weld bead metal are from the same material (ASTM A36 carbon low steel). In the end the results of fatigue analysis showed the maximum stress was reached in Butt joint (250Mpa) and in pipe-flange Butt joint(544.2Mpa), while the maximum stress on Lap joint (58.23Mpa) and the maximum stress on a Tee joint (250Mpa).

Key Words: Fatigue Analysis, Welding Joints, Low Carbon Steel, Finite Element Method , Maximum Stress.

الخلاصة

في هذه الدراسة، تم تحليل الكلال لعدة انواع من وصلات اللحام ، من ضمنها الوصلة التناكبيةButt (للصفائح والانابيب) والوصلة التراكبية Lap و وصلة اللحام الركنية من الجهتين Tee . هذه الملحومات تم دراستها تحت حمل متزايد باستخدام برنامج CATIA software للتصميم الملحومات وباستخدام برنامج ABAQUS environment لتحليل الاجهادات الناتجة. تتضمن هذه الدراسة اللحام باستخدام القوس الكهربائي. كل الاجزاء الملحومة وخرزة اللحام من نفس المادة وهي الفولاذ واطئ الكاربون . في نهاية نتائج تحليل الكلال اعلى اجهاد يظهر في وصلة الحام التناكبية للصفائح مقداره (250Mpa) ، بينما اعلى اجهاد تصل اليه الوصلة التناكبية للأنابيب مقداره(544.2Mpa) ، في حين اعلى اجهاد وصلت اليه الوصلة التراكبية (58.23Mpa) ، واعلى اجهاد وصلت اليه الوصلة الركنية من الجهتين (250Mpa).

الكلمات المفتاحية : تحليل الكلال ، وصلات اللحام ، فولاذ واطئ الكاربون ، طريقة الشرائح المحددة ، اعلى اجهاد.

1. Introduction

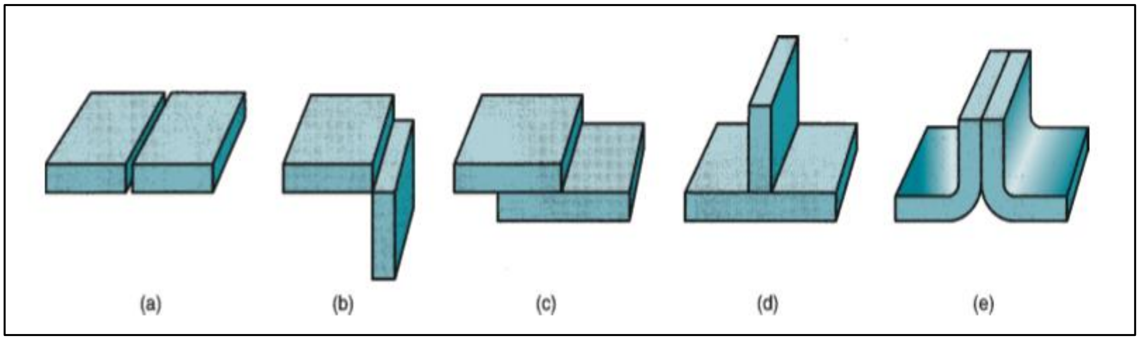
Arc welding is a group of welding processes that use an electric arc as a source of heat to melt and join metals, filler or pressure might or might not be required. These processes include: 1) Gas Tungsten Arc (GTA, TIG), 2) Gas Metal Arc (GMA, MIG, MAG), 3) Submerged Arc Welding(SAW) and 4) Shielded Metal Arc Welding (SMAW). (Khan, 2007)

The joint type selected depending on the plate thickness and welding method. Ideal joint provides the quality and required structural strength without an unnecessarily large joint volume. As the size of the joint increase, the weld cost increases, and the higher heat input will cause problems with distortion and impact strength. (Groover, 2010)

There are five basic types of weld joints for bringing two parts together.

a) Butt joint : In this type, the parts must lie in the same plane and joined at their edges.
b) Corner joint : In this joint type , in a corner joint the parts joined at the corner of the right angle (90°).

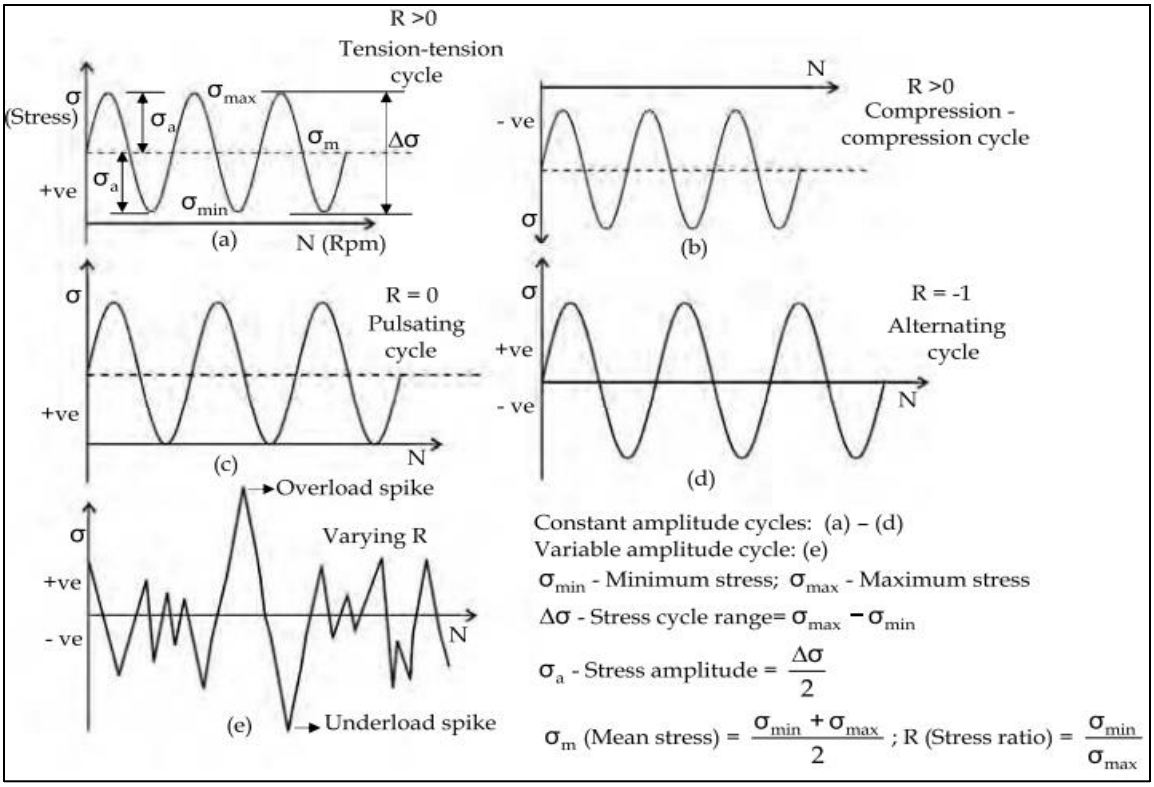
c) Lap joint : This type of joint contains two overlapping parts.
d) Tee joint : In this type, The parts are perpendicular to each other in T-shape.
e) Edge joint : In an edge joint, the parts are parallel with at least one of their edges. (Groover, 2010)



Figure(1): Types of Welded Joint. (Groover, 2010)

Two of the major welding process problems are distortion and residual stress. As the material heats and expands during welding, residual stress is primarily caused by the compressive yielding that happens around the molten zone. Then it contracts when the weld metal cools, which causes a tensile residual stress, principally in the longitudinal direction. After welding a residual tensile stress remains across the weld centerline and causes a balancing compressive stress further from the weld zone . On the weld line the tensile residual stress decreases the toughness and fatigue strength, principally when shared with any defects associated with the weld bead or notches. (Colegrove *et.al.*, 2009)

In service lives of typical structural components , the structural components are subjected to cyclic loads. These loads might have an amplitude that varies with time or a constant amplitude. For cases that show constant amplitude loading with or without mean offset loading , the determination of the amplitude of a cycle and the number of cycles experienced by a component is a straightforward exercise . On the other hand, if the amplitude of the loading change with time, it is more difficult to determine what constitutes a cycle and the corresponding amplitude of the cycle. (Hathaway *et.al.*, 2005) Rotating machines usually operate under pre-decided constant amplitude load cycles where ships and aircrafts are subjected to variable amplitude load cycles due to fluctuating and random wind or sea gusts. Figure(2) shown common types of load cycles. (Morales , 2011)



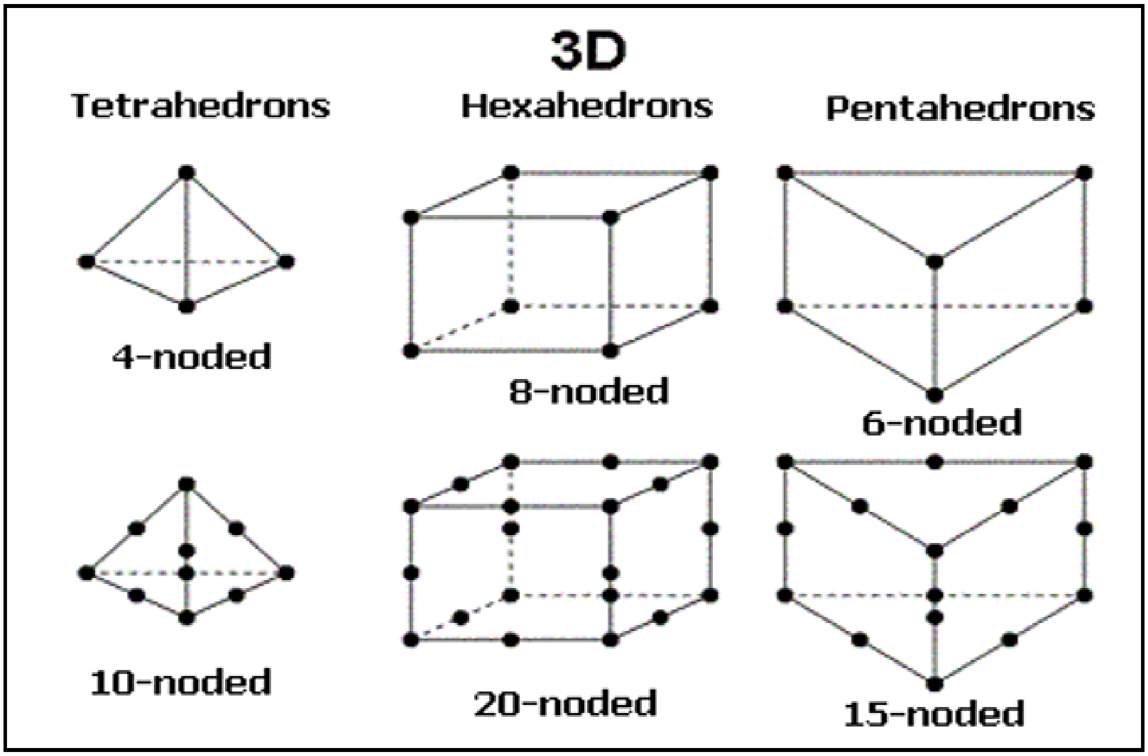
Figure(2): Types of Fatigue Cycles. (Morales ،2011)

2. Finite Element Analysis(FEM):

Finite element analysis (FEM) is a numerical analysis method for solving integral or differential equation (Dixit, 2007). It is a computational technique used to obtain approximate solutions of boundary value problems in engineering. (Hutton, 2004)

Finite element analysis, when applied to any realistic model of an engineering component, requires an enormous amount of computation and the development of the method has depended on the availability of suitable digital computers for it to run on. The method is now applied to problems including a wide range of phenomena, counting vibrations, heat conduction, fluid mechanics and electrostatics, and a wide range of material properties, such as linear-elastic behavior and behavior involving deviation from Hooke's law (for example, plasticity or rubber-elasticity) (Geoffrey , 2010).

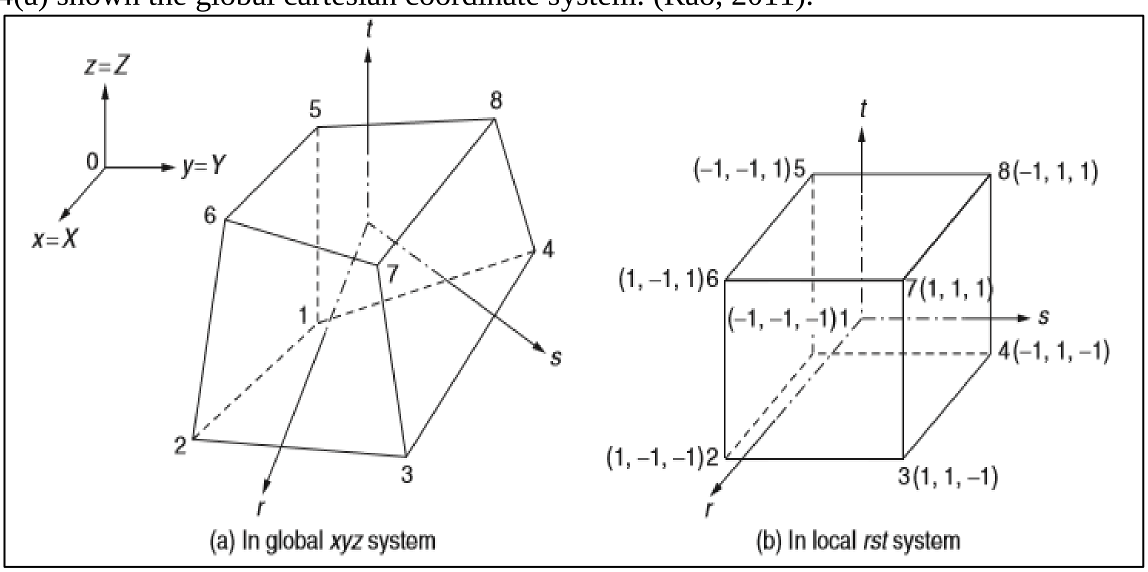
Elements can be of various shapes (as shown in figure 3), in two dimensions, quadrilateral or triangular, and in three-dimensions, brick-shaped (hexahedral), wedge-shaped (pentahedral) or tetrahedral. This is, of course, not an exhaustive list. (Geoffrey , 2010).



Figure(3): Various Finite Elements Commonly Available. (Geoffrey , 2010).

2.1.Hexahedron Element(HE):

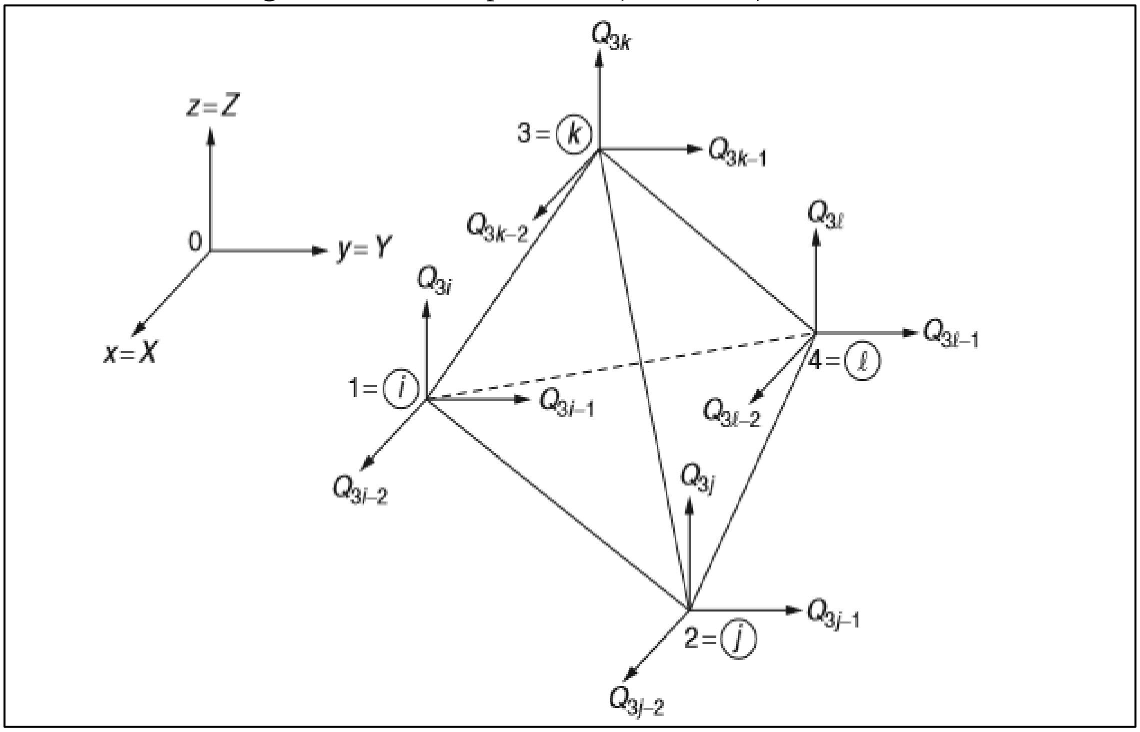
The natural coordinates are (r, s, and t) with the origin of the system occupied at the centroid of the element, as exposed in figure 4(a). Each of the coordinate axes (r, s, and t) which are given by the coordinate values (±1), associated with a pair of opposite faces. As described in figure 4(b), the element is a cube in the natural coordinates, while figure 4(a) shown the global cartesian coordinate system. (Rao, 2011).



Figure(4): Hexahedron Element (Rao, 2011).

2.2.Tetrahedron Element(TE):

As shown in Figure (5), TE displayed in the global xyz coordinate system with three translational degrees of freedom per node. (Rao, 2011).

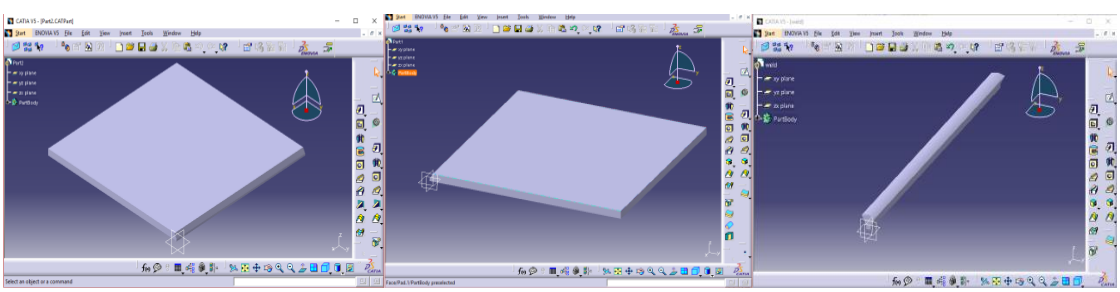


Figure(5): Tetrahedron Element (Rao, 2011).

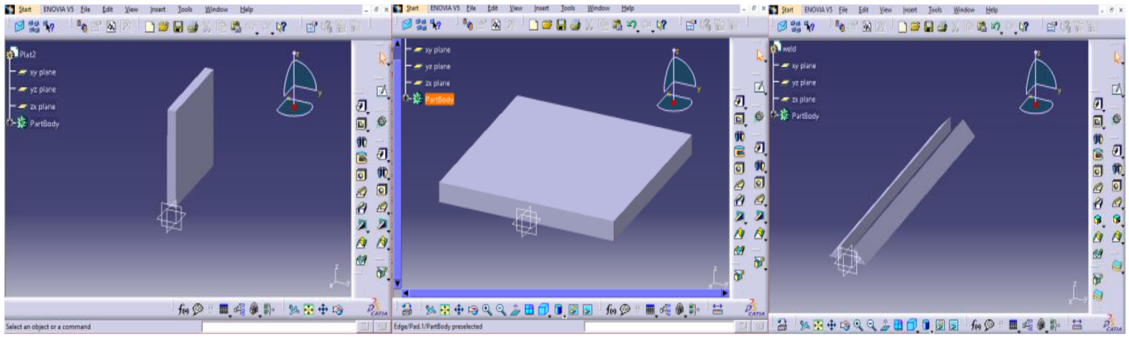
3.Finite Element Analysis and Numerical Modeling

3.1.CATIA software Modeling:

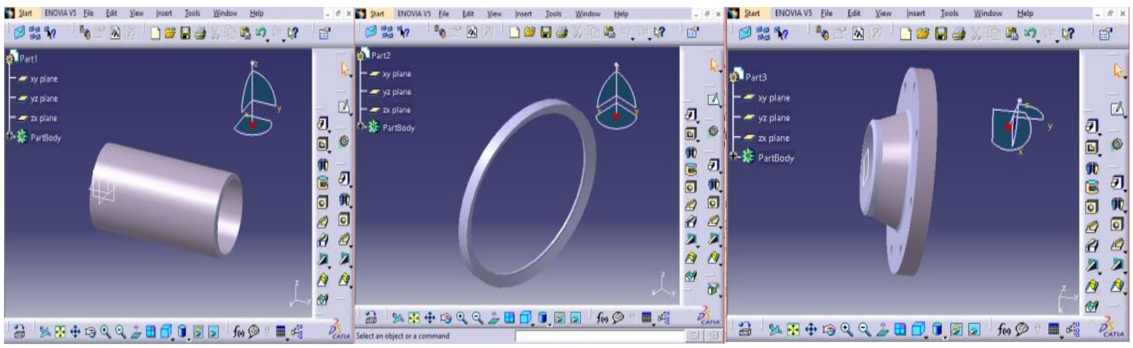
The design in CATIA program done by drawing each part separately, then the parts are assembled in the ABAQUS program. In this research three types of weld joint butt, lap and tee joint been designed each one and define the mechanical and physical properties and applied the fixing and force. Will be selected tee joint and lap joint for joining plates and selected butt joint for joining tube with flange and also for joining plates together. As shown in figure(6 ,7 ,8 and 9).



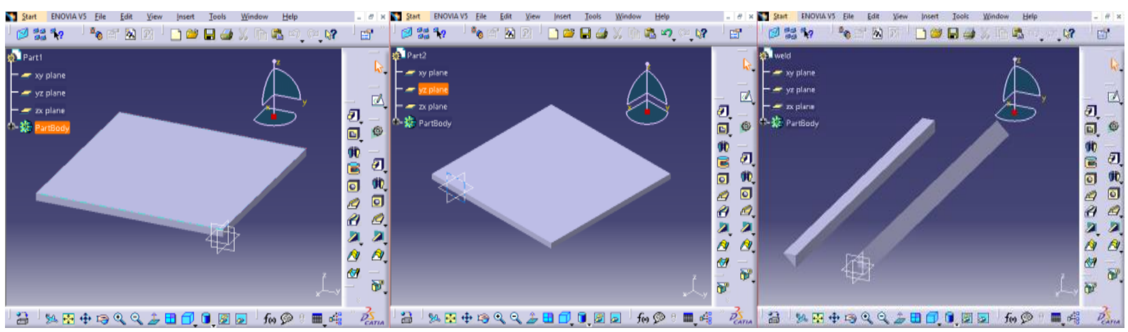
Figure(6). Parts of Butt Joint.



Figure(7). Parts of Tee Joint.



Figure(8). Parts of Pipe-Flange Butt Joint.

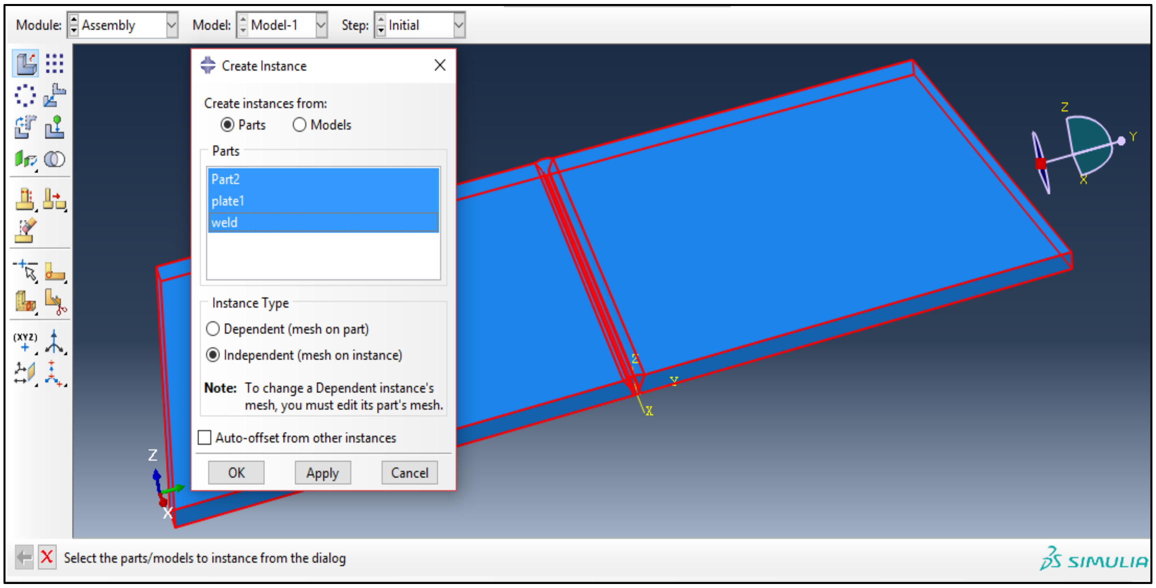


Figure(9). Parts of Lap Joint.

The dimensions for each plate was (100*100*4)all dimensions in mm , Outer and inner diameter of the tube and flange are (100mm)and (88mm) respectively. In butt joint The dimensions taken for weld pool shape is (1.5)mm in the root opining, (1.2)mm in the root face and angle of (60°) for V-joint. In pipe- flange butt joint the dimension Taken for weld pool shape was (1.2mm) in root opining ,(1.5mm)in the root face and angle of (60°) for v-joint. In tee and lap joint the dimensions taken for weld metal was (4 mm) for each side.

3.2. ABAQUS environment Analysis:

To assemble the parts as one part must go to assembly And create an instance in ABAQUS environment. A new window will appear, select all the part and chose independent mesh to create a mesh on instant not on the part. As shown in the figure(10).



figure(10): Create an Instance.

The weld metal and the parts to be joined are all of one metal, ASTM A36. is a low carbon steel that exhibits good strength coupled with formability. It is easy to machine and fabricate and can be securely welded. A36 plate can be used for a wide range of applications, depending on the thickness and corrosion resistance of the alloy.

The mechanical properties and Chemical Composition of this metal are in table(1)and (2):

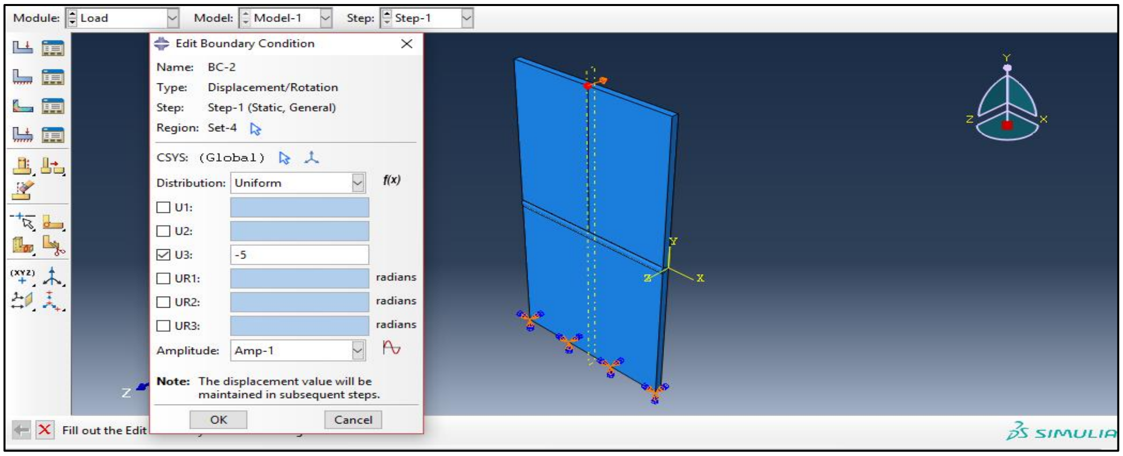
Table(1): Chemical Composition of ASTM A36.

material	Element	Composition(Max
ASTM A36	C	0.026
	Mn	0
	Si	0.4
	P	0.04
	S	0.05

Table(2): Mechanical Properties of ASTM A36.

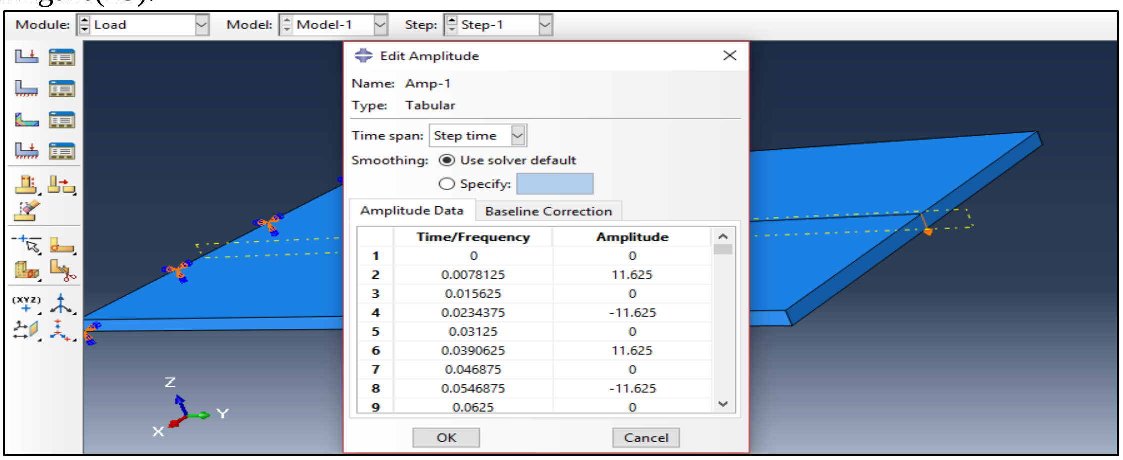
material	Tensile Test	MPa
ASTM A36	Tensile Strength	400 - 550 MPa
	Yield Strength	250 MPa
	Elongation	20.0 %

In fatigue analysis are similar to those in vibration analysis, except that in fatigue analysis can apply load at a selected point, but instead of applying force in a value form, the forces will be applied in the form of displacement with amplitude load. As shown in figure(11), and the offset will be 5mm down in Z direction.

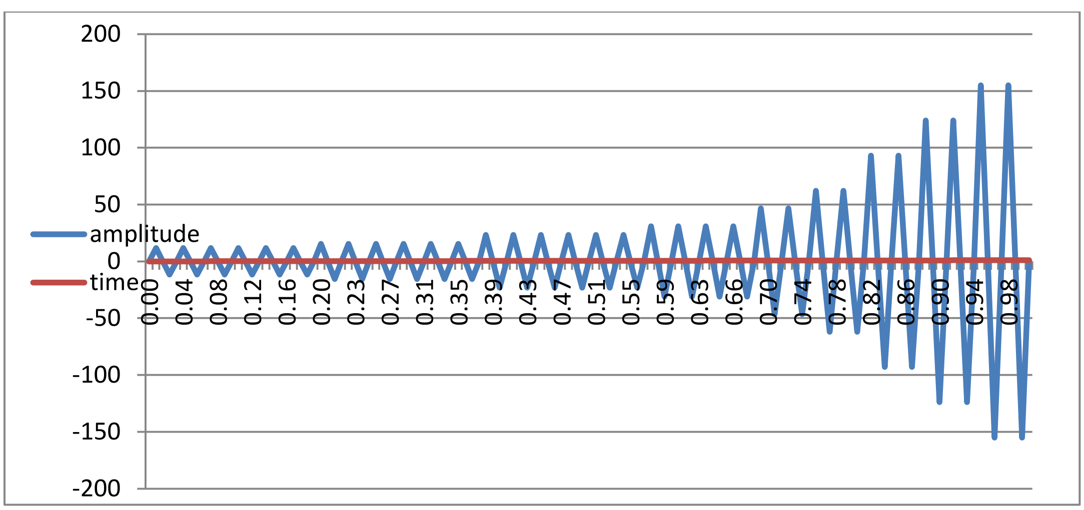


figure(11): Edit Boundary Condition (Displacement Value).

To insert load values for time periods must go to amplitude chose tabular and enter the values of loads and time as shown in figure(12). By using reversed stress as illustrated in figure(13).

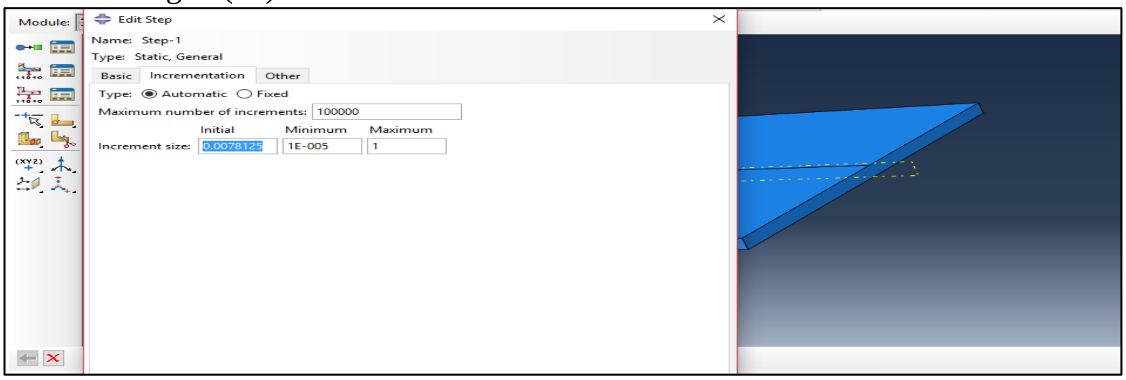


figure(12): Edit Boundary Condition (Displacement Value).



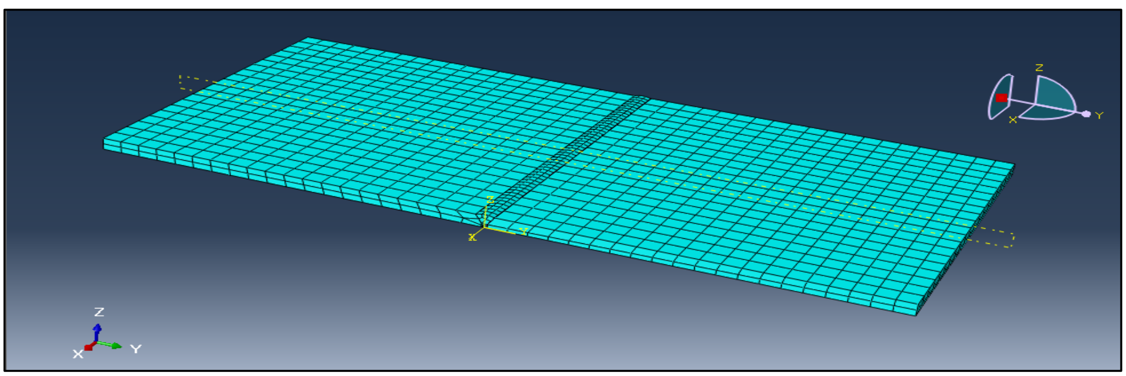
figure(13): Load Values for Time Periods.

For fatigue analysis the step is very important ,So it will create general and static step , Then enter a maximum number of increment of 100000 after that enter the initial and maximum increment size (0.0078125) and (1) respectively according to time periods, as shown in figure(14).

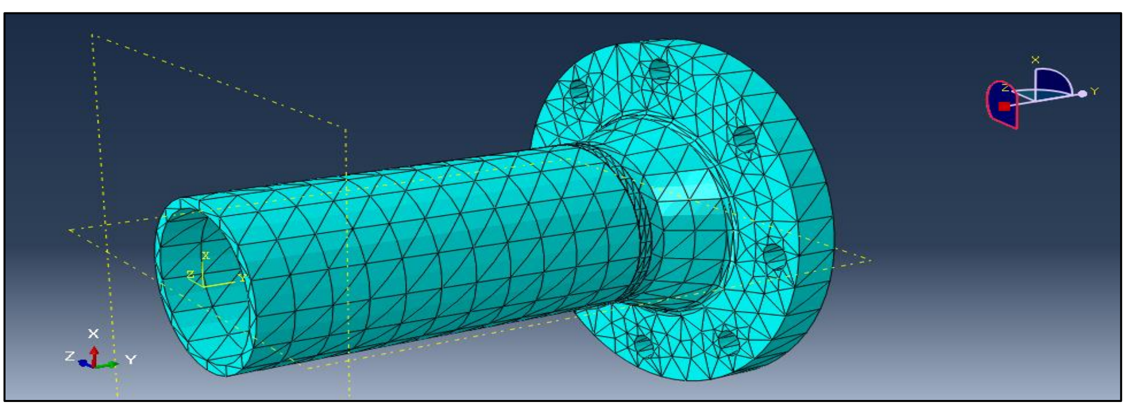


Figure(14): Edit General Static Step.

In order to make the mesh, the type of mesh must be selected from the "element type", where the type of hexagonal is chosen for plate welding (Tee, Lab and Butt Joint for Plates), while the tetrahedron is chosen for the pipe-flange butt joint, because it is irregular and cannot be meshed using hexagonal. Figure(15) illustrates the hexagonal mesh of plates, while figure16 shows tetrahedron mesh.



Figure(15): Hexagonal Mesh of Plates.

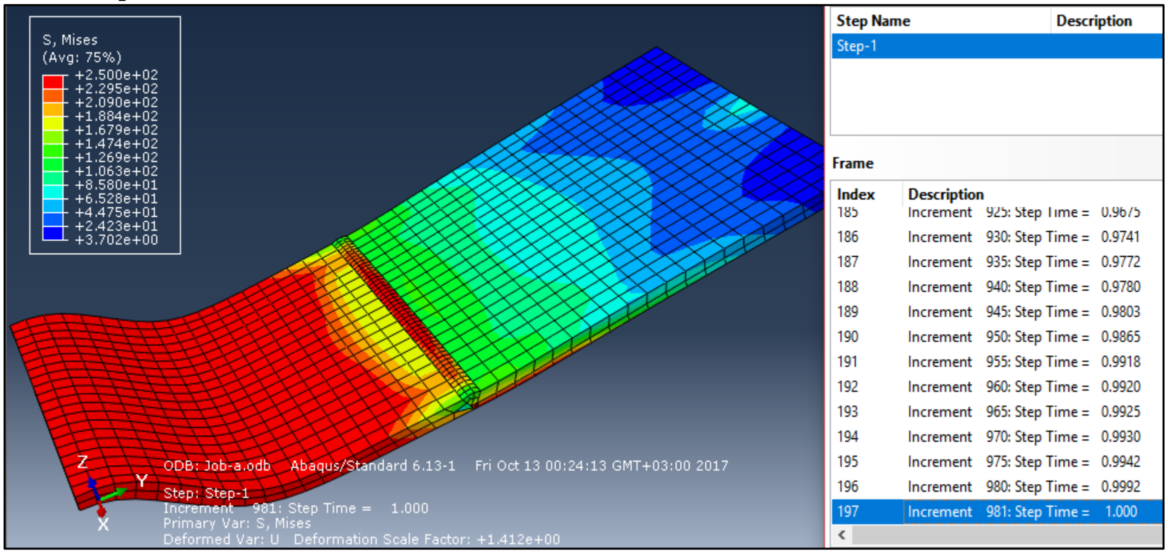


Figure(16): Tetrahedron Mesh of Pipe-Flange Butt Joint.

Finally, job is created and These steps are repeated for other joints.

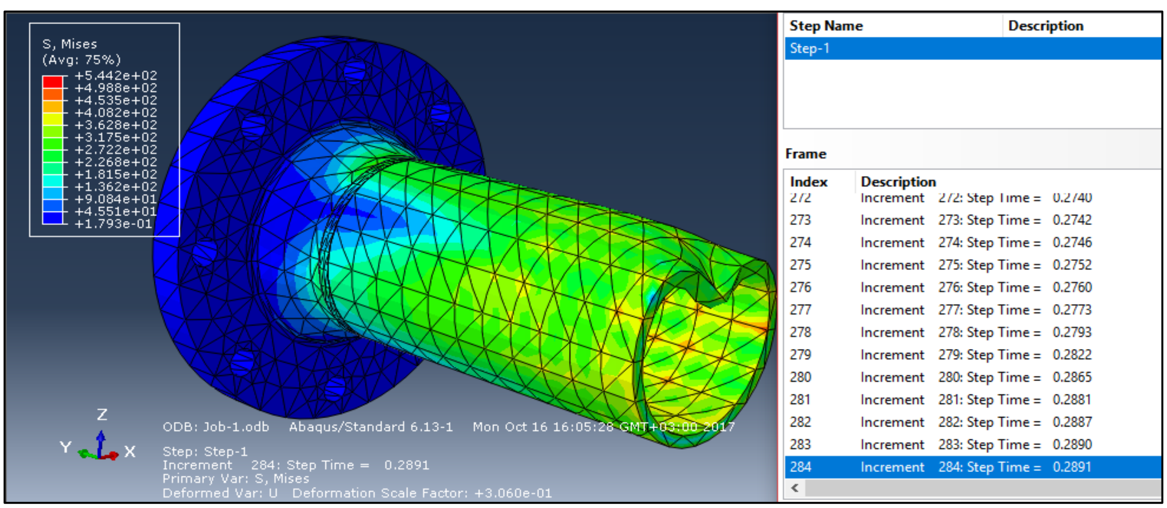
4. Result and Discussions

The results of Von Mises Stress in a butt joint completed at 197 increments and at the required time (1.00), as shown in fig.(17). The highest value of stress is (250Mpa) in the fixed plate, which is the Yield Point of A36 low carbon steel plate. Where the installed plate is made of A36 low carbon steel.



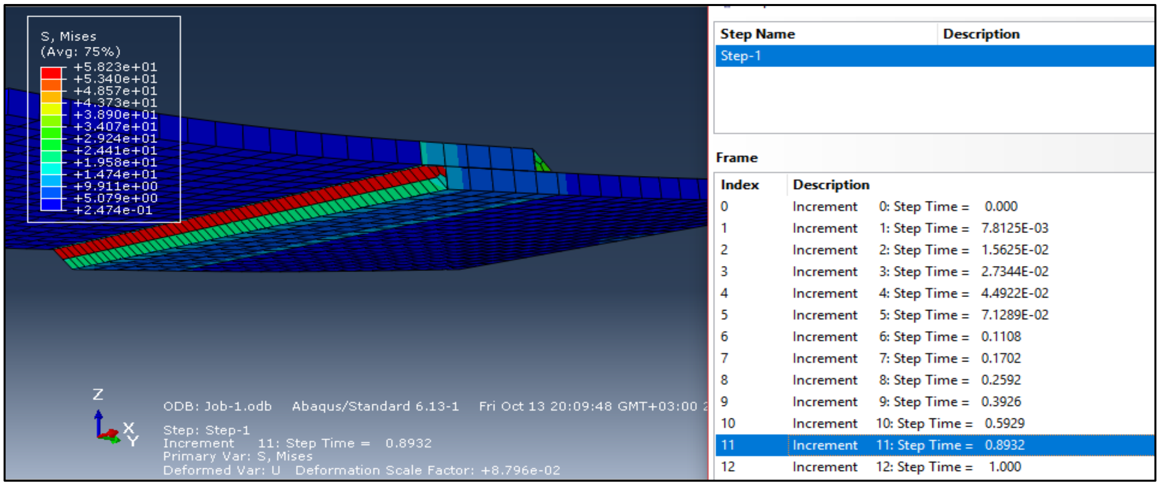
Figure(17): Von Mises Stress result of Butt Joint .

In pipe-flange butt joint, weldment analysis stops at 284 increment and at time (0.2891) where the weldment failed before reaching the required time as shown in fig.(18). The highest value of stress is (544.2Mpa) in the free end of the pipe which is made of A36 low carbon steel.



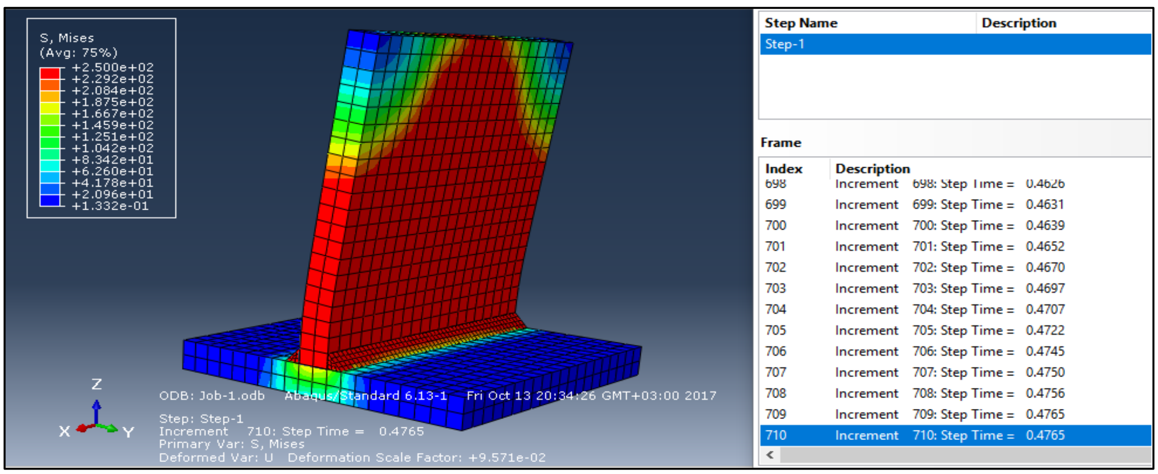
Figure(18): Von Mises Stress result of But Joint between Pipe and Flange .

In lap joint, weldment analysis is completed at 12 increment and at time (1.00), which is the required time. As shown in the figure(19), the greatest stress is at the lower welding bead of (58.23Mpa), which is below the yielding point of the welding bead made of A36 low carbon steel.



Figure(19): Von Mises Stress result of Lap Joint .

The weldment analysis is stopped in Increment 710 and at time (0.4765), as shown in figure(20), which is less than the time required for completion the analysis due to the arrival of the upper plate made of A36 low carbon steel to the yield point at this time. The maximum stress is on the vertical plate of (250Mpa).



Figure(20): Von Mises Stress result of Tee Joint.

5. Conclusions

1. ABAQUS environment shows accurate result of fatigue and dynamic analysis of the weld joint in Arc Welding.
2. In the fatigue analysis, the pipe-flange Butt joint was subjected to higher stress than other joints, then followed by Tee joint , then Butt joint, and finally Lap joint was subjected to less stress than other joints.
3. In Butt joint, the maximum stress concentration in the fixed plat of(250Mpa).
4. In pipe-flange Butt joint, the higher stress at the free end of the tube(544.2Mpa).
5. In Lap joint, the maximum stress concentration in the weld bead of (58.23Mpa).
6. In Tee joint, the maximum stress concentrate in the vertical plat of (250Mpa).

7. The fatigue analysis is stopped in all joints before reaching the desired time, except for Butt and Lap joint.

6. Reference

Colegrove, Paul *et.al.*,2009 ," *The welding process impact on residual stress and distortion*"s.l. : Science and Technology of Welding and Joining, Vol. 14, pp. 717-725.

Dixit Uday S ,2007, "*Finite Element Method".* India : Springer.

Geoffrey Modlen ,2010 , "*Introduction to Finite Element Analysis".* Manchester : University of Manchester .

Groover, Mikell P ,2010, "*Fundamentals of Modern Manufacturing Materials,Processes,and Systems"*. United States of America : John Wiley & Sons.

Hathaway, Richard B.; Yung-Li Lee; Jwo Pan ; Barkey, Mark E, 2005 , " *Fatigue Testing and Analysis".* London : Elsevier Butterworth-Heinemann.

Hutton, David V,2004 , "*Fundamentals of Finite Element Analysis",.* New York : McGraw−Hill Copanies.

Khan, Md.Ibrahim. 2007 , "*Welding Science And Technolgy"*. New Delhi : New Age International.

Morales, Eduardo Valencia, 2011 , " *Alloy Steel* - *Properties and Use".* China : InTech.

Rao, Singiresu S. ,2011 , "*The Finite Element Method in Engineering"*. Boston : Elsevier .